**FEAMAC Example 2**

*Description***:**

This is another simple FEAMAC problem involving the same finite element mesh as FEAMAC Example 1; two unit 3D elements. This problem, however, imposes a stress free cool down from 900 °C to 23 °C (loading step one) before imposing a prescribed displacement of 0.03 in. in the 1-direction (loading step two). This stress free cool down simulates the cool down of the composite material after consolidation at elevated temperature and allows the development of residual stresses in the composite. Both elements are again assigned the same MAC/GMC material, a 33% volume fraction SiC/Ti composite with a simple 2×2 GMC repeating unit cell, thus all eight integration points in both elements have their material constitutive response described by MAC/GMC. The viscoplastic material response of the Ti is modeled via the GVIPS constitutive model within MAC/GMC [2]. This FEAMAC problem is again identical to a stand-alone MAC/GMC problem in which a stress free cool down is followed by the application of a strain of 0.015 to the composite. Via commenting and un-commenting several lines in the ABAQUS and MAC/GMC input files, both longitudinal loading (in the fiber direction) and transverse loading (normal to the fiber direction) are considered in separate executions.

*Required Files:*

The following files should be placed in the ABAQUS working directory:

| File | Purpose |
|------|---------|
| feamac_ex2.inp | ABAQUS input file |
| SiC-Ti_33b.mac | MAC/GMC input file describing the SiC/Ti composite material |
| feamac.for | User-defined subroutines for FEAMAC |

*Execution:*

This problem can be executed via the following command at the ABAQUS command line:

```
abaqus –j feamac_ex2 –user feamac interactive
```

The `–j` specification indicates the job name (i.e., ABAQUS input file name), while the `–user` specification indicates the file containing the FEAMAC user-defined subroutines. The `interactive` specification provides detailed information on the problem execution during the execution and is optional.

*Output:*

The output for this problem is written to the ABAQUS output database file `feamac_ex2.odb` for post-processing in ABAQUS/CAE, ABAQUS/Viewer, or other appropriate finite element post-processing software. In addition, MAC/GMC output is written to 32 ASCII files named `SIC-TI_33b_macro_EL`*i*`_PT`*j*`.data` and `SIC-TI_33b2_macro_EL`*i*`_PT`*j*`.data`, where *i* and *j* are the element number and integration point number in the ABAQUS finite element model, respectively. These files contain the macro (repeating unit cell) level output specified in the MAC/GMC input file. "`_macro`" and "`_EL`*i*`_PT`*j*`.data`" have been appended to the name specified for the macro output in the MAC/GMC input file. Finally, a MAC/GMC output file is written for each MAC/GMC input file employed in the ABAQUS model. In this case this file is `SiC-Ti_33b.out`. It contains an echo of the MAC/GMC input file data and results in the form of effective properties. If an error is found in the MAC/GMC input, a message describing the error will be written to this output file.

*Results:*

Results from the cool down loading step, in the form of strain vs. temperature curves, are plotted in Fig. 6. The curves labeled "FEAMAC" are plotted from the ASCII output in the files `SIC-TI_33b_macro_EL`*i*`_PT`*j*`.data`, which are all identical due to the uniform state of stress and strain that results in the model. Because the problem simulates uniform stress and strain throughout the model, the cool down results are identical to a stand-alone MAC/GMC problem simulating stress free cool down of the SiC/Ti composite. This fact is illustrated in Fig. 6 as the results of stand-alone MAC/GMC analyses are plotted as well. The composite exhibits less strain in the longitudinal case compared to the transverse case due to the presence of the continuous low thermal expansion, stiff fibers in the longitudinal case. Figure 7 shows the predicted longitudinal and transverse mechanical stress strain curves generated by FEAMAC and MAC/GMC after the cool down loading step. Note that the starting strain for these curves has been shifted to zero from the non-zero global thermal strain that results at the end of the stress free cool down step (see Fig. 6). Figure 8 compares these stress-strain curves with residual stresses to the stress-stress strain curves without residual stresses (generated in FEAMAC Example 1). These results indicate that the inclusion of residual stresses causes an earlier onset of yielding and also a slight alteration in the post-yield slope of the stress-strain curves. Figures 9 – 11 show the total, thermal, and inelastic strain component fields (in the 1-direction) at the end of the cool down loading step for the longitudinal fiber orientation. The total strain (-0.005092) is equal to the sum of the thermal strain (-0.005718) and the inelastic strain (0.0006257). It is interesting to note that the slopes of the curves plotted in Fig. 6 do not provide the composite effective coefficient of thermal expansion (CTE). The CTE is defined as the slope of the thermal strain vs. temperature curve, not the total strain vs. temperature curves given in Fig. 6. The presence of inelasticity causes the observed difference between the total and thermal strains. The practice of determining a material's CTE via a measured total strain vs. temperature curve in the presence of inelasticity (or damage) is thus incorrect.
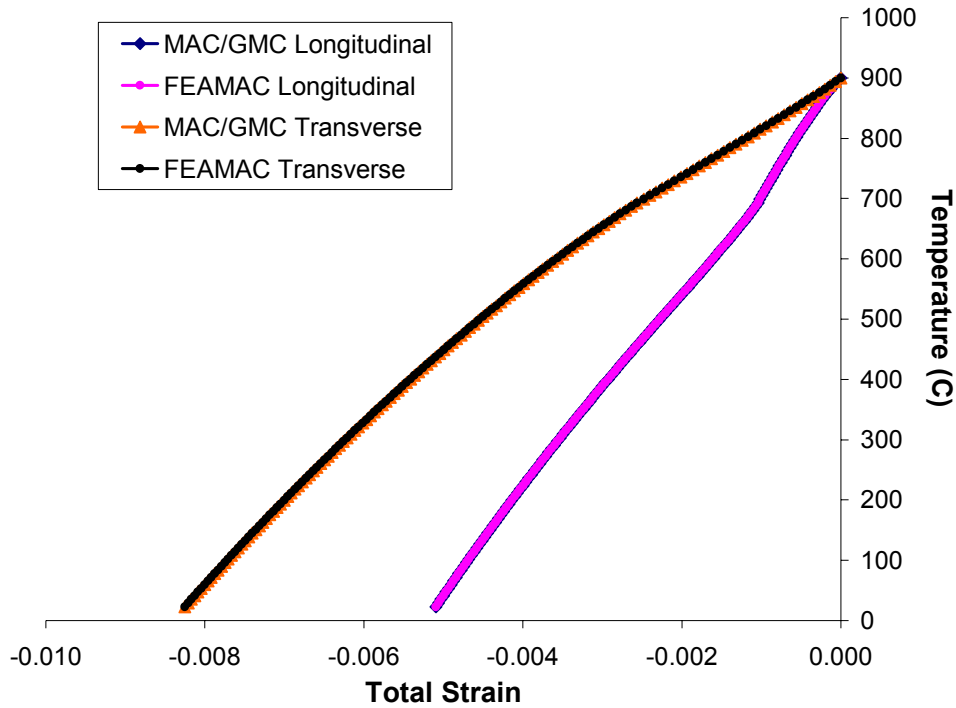


**Fig. 6**. Longitudinal and transverse strain curves during stress free cool down for a 33% SiC/Ti composite represented by a 2×2 GMC repeating unit cell predicted by stand-alone MAC/GMC and FEAMAC.
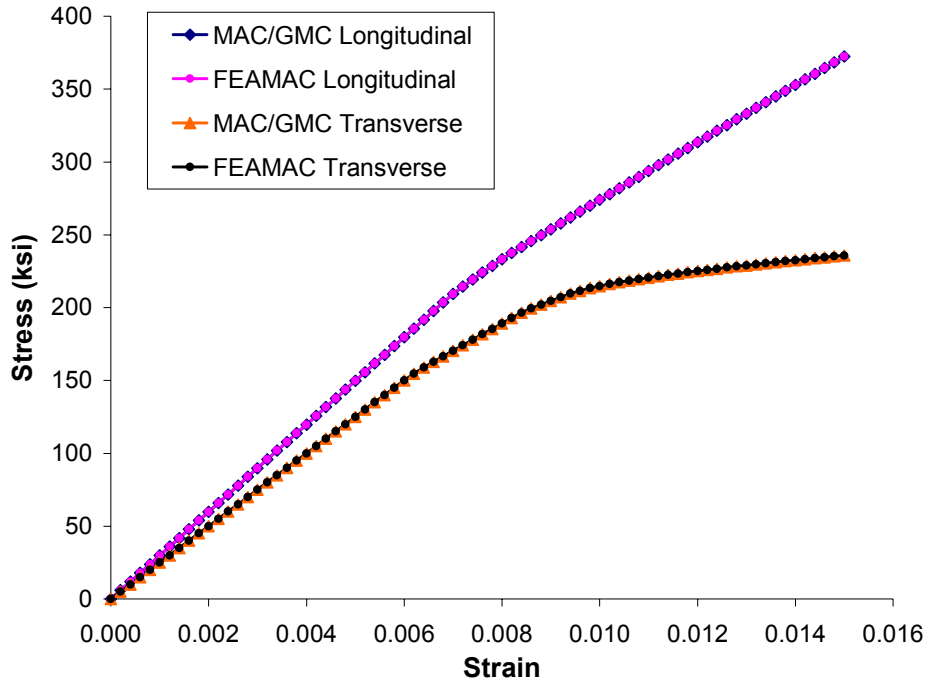
**Fig. 7**. Longitudinal and transverse stress-strain curves for a 33% SiC/Ti composite represented by a 2×2 GMC repeating unit cell predicted by stand-alone MAC/GMC and FEAMAC. Residual stresses have been incorporated via a stress free cool down prior to application of the mechanical loading.
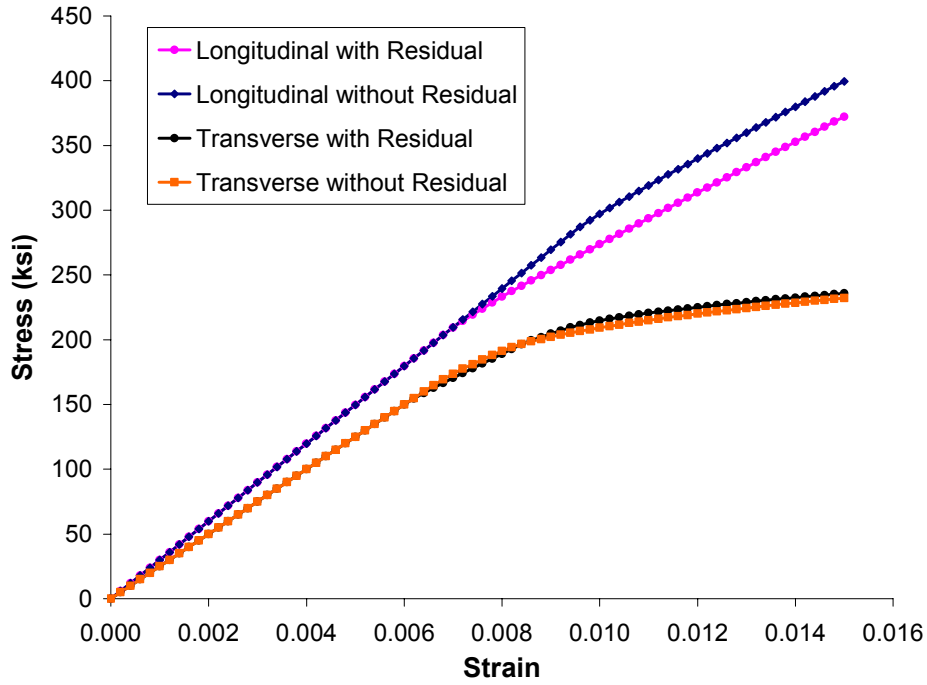


**Fig. 8**. Comparison of longitudinal and transverse stress-strain curves for a 33% SiC/Ti composite represented by a 2×2 GMC repeating unit cell predicted by FEAMAC both with and without residual stresses included.
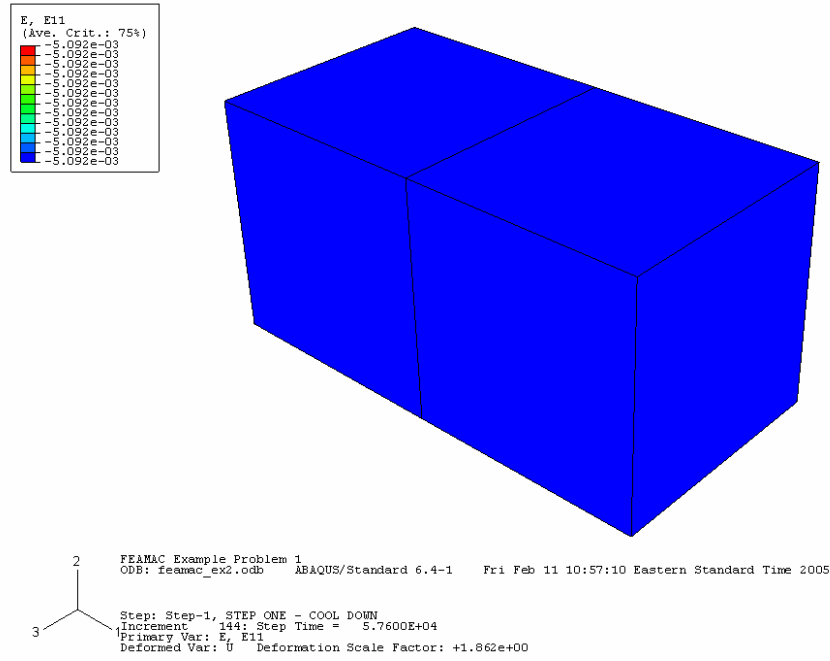
**Fig. 9**. $\varepsilon_{11}$ total strain field at the end of the cool down step in the 2 elements comprising the ABAQUS model as predicted by FEAMAC. The fiber orientation is longitudinal (in the 1-direction) in this case.



**Fig. 10**. $\varepsilon_{11}^{th}$ thermal strain field at the end of the cool down step in the 2 elements comprising the ABAQUS model as predicted by FEAMAC. The fiber orientation is longitudinal (in the 1-direction) in this case.
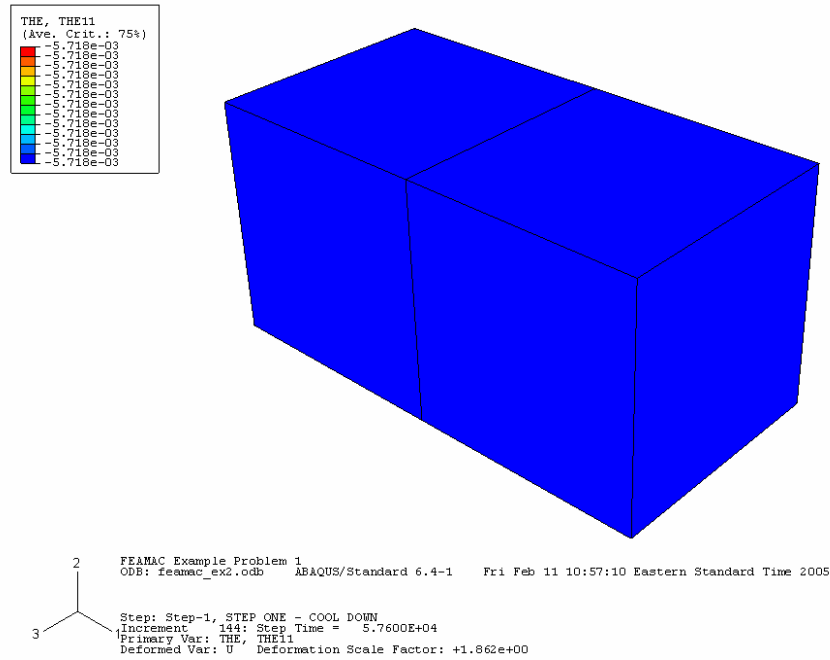
**Fig. 11**. $\varepsilon_{11}^{in}$ inelastic strain field at the end of the cool down step in the 2 elements comprising the ABAQUS model as predicted by FEAMAC. The fiber orientation is longitudinal (in the 1-direction) in this case.